SPH modeling of supercavity induced by underwater high speed objects

M. B. Liu[1,2]*
[1]Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190, China
[2]State Key Laboratory of Hydraulics and Mountain River Engineering, Sichuan University, Chengdu 610065, China

J. R. Shao
Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190, China

Z. Shang
Institute of High Performance Computing
Agency for Science, Technology and Research
#16-16 Connexis
Fusionopolis, Fusionopolis Way, Singapore 138632

SUMMARY

This paper presents a numerical simulation of supercavity induced by underwater high speed objects using smoothed particle hydrodynamics (SPH). SPH is a Lagrangian, meshfree particle method. It has special advantages in modeling free surfaces, moving interfaces and deformable boundaries. Based on Navier-Stokes equations of compressible fluids and a virtual single phase equation of state, it is feasible to apply the SPH method to bubble cavitation and super cavitations.

INTRODUCTION

For underwater high speed moving objects, cavitation happens when water moves at an extremely high speed, which causes local water pressure around the moving objects to drop below saturation pressure, creating bubble cavities. Supercavitation occurs if a bubble cavity is large enough to envelop the object and is also strong enough to maintain its integrity. As such, it is possible to greatly reduce the drag of an underwater body, and to enable it moving in a dramatic high speed underwater. This causes the rapidly development of underwater supercavitation weapons and vehicles [1-2].

Cavitation induced by high speed moving underwater objects is a complex unsteady and discontinuous or periodic phenomenon with the formation, growth and rapid collapse of bubble cavities, and therefore it is neither reliably assessable nor fully understood yet. Despite the great advances during the last decades, Cavitation and supercavitation are still an ongoing research area, while needs wider and deeper exploration of inherent mechanics.

With the advancement of computer hardware and software, computer modeling with CFD techniques has gradually become a strong tool for understanding cavitations. In this paper, we used a meshfree method, smoothed particle hydrodynamics (SPH), to numerical simulate supercavity induced by underwater high speed objects. The object of the research is to examine the shape of the supercavity, and to explore the feasibility of applying SPH to supercavity problems.

SPH METHOD AND GOVERNING EQUATIONS

SPH is a Lagrangian, particle-based meshfree method [3-5]. In SPH, the state of a system is represented by a set of particles, which possess material properties and interact with each other within the range controlled by a weight function or smoothing function. An important advantage of SPH is that in SPH, there is no explicit interface tracking for multiphase or free surface flows – the motion of the fluid is represented by the motion of the particles, and fluid surfaces or fluid-fluid interfaces move with particles representing their phase defined at the initial stage. Therefore, SPH is attractive in modeling cavity formation and evolution process.

A. SPH methodology

In conventional SPH method, the values of a particular variable at any point can be obtained using following equations:

$$< f(\mathbf{x}_i) >= \sum_{j=1}^{N} \frac{m_j}{\rho_j} f(\mathbf{x}_j) W(\mathbf{x}_i - \mathbf{x}_j, h) , \quad (1)$$

$$< \nabla \bullet f(\mathbf{x}_i) >= \sum_{j=1}^{N} \frac{m_j}{\rho_j} f(\mathbf{x}_j) \nabla_i W_{ij} , \quad (2)$$

where, $< f(\mathbf{x}_i) >$ is the approximated value of particle *i*, $f(\mathbf{x}_j)$ is the value of $f(\mathbf{x})$ associated with particle *j*, $\mathbf{x}_i$ and $\mathbf{x}_j$ are the positions of corresponding particles. m and ρ denote mass and density respectively. h is the smooth length, N is the number of the particles in the support domain. W is the smoothing function, and represents a weighted contribution of particle *j* to particle *i*. The smoothing function is sometimes referred to as kernel or kernel function, and it should satisfy some basic requirements, such as normalization condition, compact supportness, and Delta function behavior.

B. SPH equations of motion

For viscous hydrodynamic problems, the Lagrangian form governing Navier-Stokes (N-S) equation can be written as

Proceedings of the Eighth International Symposium on Cavitation (CAV 2012)
Edited by Claus-Dieter OHL, Evert KLASEBOER, Siew Wan OHL, Shi Wei GONG and Boo Cheong KHOO.

ISBN: 978-981-07-2826-7 :: doi:10.3850/978-981-07-2826-7_056

$$\frac{d\rho}{dt} = -\rho\nabla \bullet \mathbf{v} \,, \tag{3}$$

$$\frac{d\mathbf{v}}{dt} = -\frac{1}{\rho}\nabla p + \frac{\mu}{\rho}\nabla^2 \mathbf{v} + \mathbf{g} \,, \tag{4}$$

$$\frac{de}{dt} = \frac{\sigma^{\alpha\beta}}{\rho}\frac{\partial \mathbf{v}^{\alpha}}{\partial \mathbf{x}^{\beta}} \,, \tag{5}$$

where $\mathbf{v}, p, \mathbf{g}$, e and μ denote velocity vector, pressure, gravity, internal energy and dynamic viscosity respectively. The total stress tensor $\sigma^{\alpha\beta}$ in equation (5) is made up of two parts, one part of isotropic pressure p and the other part of viscous stress τ

$$\sigma^{\alpha\beta} = -p\delta^{\alpha\beta} + \tau^{\alpha\beta} \,, \tag{6}$$

where $\delta_{\alpha\beta} = 1$ if $\alpha = \beta$, and $\delta_{\alpha\beta} = 0$ if $\alpha \neq \beta$.

The pressure can be calculated through an equation of state, $p = p(\rho, e)$. For Newtonian fluids, the viscous stress should be proportional to the strain rate denoted by ε through the dynamic viscosity μ .

$$\tau^{\alpha\beta} = \mu\varepsilon^{\alpha\beta} \,, \tag{7}$$

where

$$\varepsilon^{\alpha\beta} = \frac{\partial \boldsymbol{v}^{\beta}}{\partial \boldsymbol{x}^{\alpha}} + \frac{\partial \boldsymbol{v}^{\alpha}}{\partial \boldsymbol{x}^{\beta}} - \frac{2}{3}(\nabla \cdot \boldsymbol{v})\delta^{\alpha\beta} \,. \tag{8}$$

According to equations (1) and (2), the SPH equations of motion for the N-S equation can be obtained as follows

$$\frac{d\rho_i}{dt} = \sum_{j=1}^{N} m_j \mathbf{v}_{ij} \nabla_i W_{ij} \,, \tag{9}$$

$$\frac{d\mathbf{v}_i}{dt} = -\sum_{j=1}^{N} m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2}\right) \bullet \nabla_i W_{ij} + \sum_{j=1}^{N} \frac{4 m_j \left(\mu_i + \mu_j\right) \mathbf{x}_{ij} \bullet \nabla_i W_{ij}}{\left(\rho_i + \rho_j\right)^2 \left(x_{ij}^{\ 2} + 0.01h^2\right)} \boldsymbol{v}_{ij} + \mathbf{g} \,, \tag{10}$$

$$\frac{de_i}{dt} = \frac{1}{2}\sum_{j=1}^{N} m_j \frac{p_i + p_j}{\rho_i \rho_j} \boldsymbol{v}_{ij}^{\beta} \frac{\partial W_{ij}}{\partial \boldsymbol{x}_i^{\beta}} + \frac{\mu_i}{2\rho_i} \varepsilon_i^{\alpha\beta} \varepsilon_i^{\alpha\beta} \,, \tag{11}$$

For hydrodynamic problems associated with shock waves, an artificial viscosity is usually used in SPH method to stabilize the numerical scheme, prevent particle penetration, diffuse sharp variation, and capture shock waves. Shock wave is generated when liquid drop impacting onto liquid films or solid walls, with the magnitude depending on the impacting velocity. In this paper, Monaghan type artificial viscosity [6] (Π_{ij}) is added to equation (10) in the physical pressure term.

It is noted that when underwater objects move in a low speed, water can be considered as incompressible. In contrast, when the underwater objects move in a high speed, compressibility of water must be considered, especially for water close to the moving objects. For the previous case, the SPH method usually uses an artificial compressibility technique to model the incompressible flow as a slightly compressible flow. The artificial compressibility considers that every theoretically incompressible fluid is actually compressible. Therefore, it is feasible to use a quasi-incompressible equation of state to model the incompressible flow. The purpose of introducing the artificial compressibility is to produce the time derivative of pressure. A possible artificial equation of state is

$$p = c^2 \rho \,, \tag{12}$$

where c is the sound speed which is a key factor that deserves careful consideration. For the later case of highly compressible flow, a true equation of state which can take account of the compressibility of water must be used. In this work, the Mie-Gruneisen equation of state was used. The pressure of water in the compression state is given by

$$p = \frac{\rho_0 C^2 \psi [1 + (1 - \frac{\gamma_0}{2})\psi - \frac{a}{2}\psi^2]}{[1 - (S_1 - 1)\psi - S_2 \frac{\psi^2}{\psi + 1} - S_3 \frac{\psi^3}{(\psi + 1)^2}]^2} + (\gamma_0 + a\psi)e \,, \tag{13}$$

In the case of expansion, the pressure of water is

$$p = \rho_0 C_0^2 \psi + (\gamma_0 + a\psi)e \,. \tag{14}$$

where ρ_0 is the initial density, η is the ratio of the density after and before disturbance, and $\psi = \eta - 1$. When $\psi > 0$, water is in compressed state, and when $\psi < 0$, water is in expanded state. Some material parameters and coefficients of the Mie-Gruneisen equation of state for water are given as follows

Table 1 Material parameters and coefficients of the Mie-Gruneisen equation of state for water

Symbol	Meaning	Value
ρ_0	Initial density	1000 Kg/m^3
C_0	Reference sound speed	1480 m/s
γ_0	Gruneisen coefficient	0.5
a	Volume correction coefficient	0
S_1	Fitting coefficient	2.56
S_2	Fitting coefficient	1.986
S_3	Fitting coefficient	1.2268

In our work, The SPH model involves three major modifications on the traditional SPH method, 1) correction on the SPH kernel and kernel gradients to improve the computational accuracy in particle approximation, 2) RANS turbulence model to capture the inherent physics of flow turbulence, and 3) enhanced solid boundary treatment algorithm with pressure correction to remove fake pressure oscillation near solid boundaries. Detailed information regarding to the SPH methodology can be obtained from [3-5].

NUMERICAL VALIDATION

In order to validate the SPH method in modeling water entry and exit, the water entry of a cylinder is simulated as a benchmark problem. As shown in Figure 1, the length and height of the water are 200 mm and 50 mm, respectively, and the radius of the cylinder is 5.5 mm, which has the same density as water. The initial downward velocity of the cylinder is 2.955 m/s.

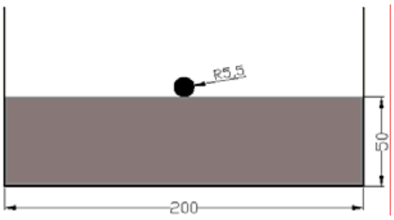


Figure 1: Numerical model of the water entry of a cylinder (unit: mm).

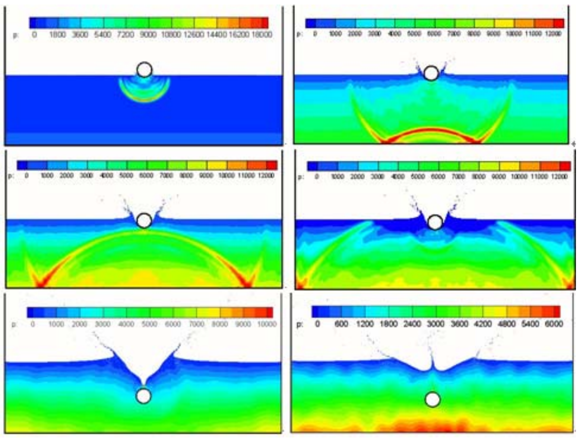


Figure 2: Pressure evolution at 0.006, 0.02 0.03, 0.035, 0.2, 0.26 s.

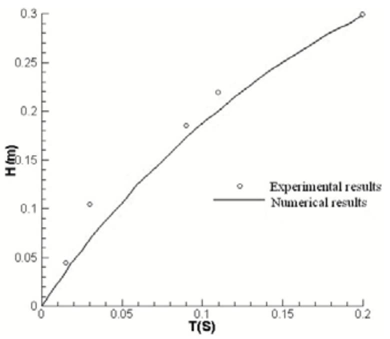


Figure 3: Penetration depths obtained from numerical simulation and experimental observation.

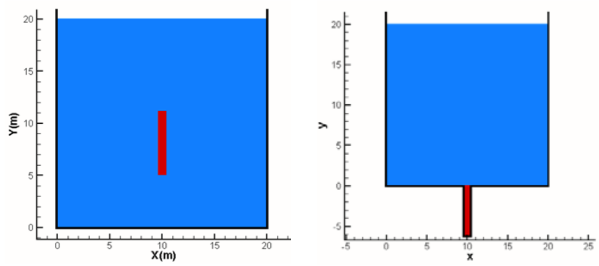
Figure 2 shows the pressure evolution during the water entry process at 0.006, 0.02 0.03, 0.035, 0.2, 0.26 s. At 0.006 s when the cylinder meets the water surface, a pressure wave produces and then transmits in water. At 0.02 s, after the interaction with the solid wall, the pressure wave changes its direction and forms a reflection wave with a maximum pressure of about 12000 Pa. The whole pressure field is smooth during the pressure wave propagation. The reflection wave meets the falling cylinder at about 0.03 s, and produces a new interaction with the cylinder. With these disturbances, the pressure field is not as smooth as before. However, this effect will gradually disappear as time elapses and the pressure field becomes smooth again.

Figure 3 shows the experimental observations by Greenhow [7] and SPH results. It is clear that the obtained SPH results are close to experimental observations, and it shows that the SPH method is valid for problems related to water entry and exit.

SUPERCAVITY INDUCED BY UNDERWATER OBJECTS

A. Problem setup

In this section, two cases of water exit of a projectile (or projectile launch) are simulated. One case is associated with projectile launching right from water (Figure 4a), while the other case is projectile launching from a launch canister (Figure 4b). The initial launching speed is 100 m/s, which is a subsonic launch.



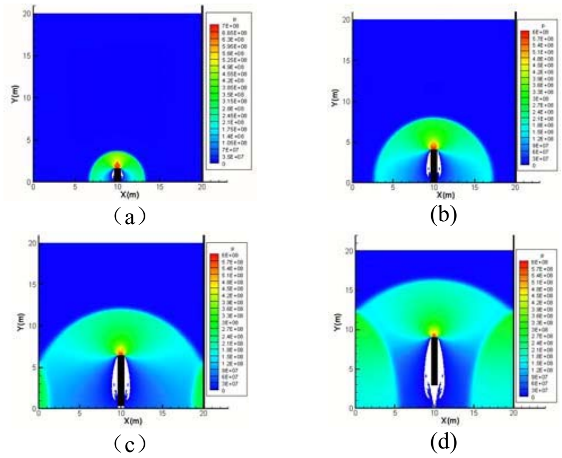
(a) (b)

Figure 4: Illustration of problem setup.

B. Results and discussions

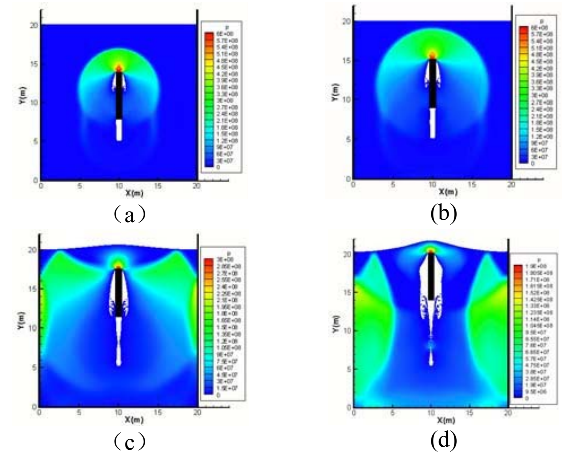
Figure 5 shows the pressure evolution during projectile launching right from water at 0.0016, 0.004, 0.0064, and 0.009 s. It is clear that right after launching the projectile, a pressure wave is generated, with a cavity formed around the projectile. After then with the cruise of the projectile, pressure wave propagates in water while the bubble cavity gradually grows up. Once the pressure wave reaches the solid wall, it reflects backwards into water region, interplaying with outward propagating pressure wave, and influencing the movement of the projectile.

Figure 6 shows the pressure evolution during projectile launching from a launch canister at 0.0028, 0.004, 0.0064 and 0.009 s. Similar to projectile launching from water, pressure wave generation, propagation, reflection and interaction can also be observed. It is noted that when the projectile is launched from water, a bubble cavity is generated, and developed bigger to gradually envelop the projectile. In contrast, when the projectile is launched from a canister, there are two bubble cavities, a main bubble cavity enveloping the projectile, and a smaller one around the aft of projection. The circulatory flow is also more obvious.



(a) (b) (c) (d)

Figure 5: Pressure evolution during projectile launching right from water at 0.0016, 0.004, 0.0064, and 0.009 s.



(a) (b) (c) (d)

Figure 6: Pressure evolution during projectile launching from a launch canister at 0.0028, 0.004, 0.0064 and 0.009 s.

REMARKS AND CONCLUSIONS

This paper presents an SPH modeling of projectile launching right from water and from a launch canister. The meshfree, particle Lagrangian method is appealing in modeling cavity flows induced from underwater high speed moving objects. For both cases, inherent physics of pressure wave generation, propagation, reflection and interaction as well as bubble cavity formation and evolution can be well described.

It is noted that current setups of projectile are limited to size of the geometry, with which the reflection wave can have significant influence on the dynamics of projectile. For modelling realistic projectile launching, a bigger computational domain or a wave-damping technique is necessary. Also in the presented SPH model, cavitations criterion is not considered. This is acceptable for predicting cavity shape, but is not sufficient in providing interaction of liquid and vapor phases. Future work will also need incorporation of a reliable cavitation model, and need to take account of the collapse of bubble cavity when the projectile exits water..

ACKNOWLEDGMENTS

This work has been supported by the National Natural Science Foundation of China (11172306) and National Defense Innovation Funds of the Chinese Academy of Sciences (Y175031XML).

REFERENCES

[1] McCormick, B. W. 1962, "On cavitation produced by a vortex trailing from a lifting surface," *Trans. ASME, J. of Basic Engineering*, 84, Series D, No. 3, 369-379.

[2] Physical and numerical modelling of unsteady cavitation dynamics. *Proc. 4th International Conference on Multiphase Flow,* New Orleans, USA.

[3] Monaghan J. J. 2005, "Smoothed particle hydrodynamics," *Reports on Progress in Physics,* 68(8):1703-1759.

[4] Liu M. B., Liu G. R. 2010, "Smoothed particle hydrodynamics (SPH): An overview and recent developments," *Archives of Computational Methods in Engineering,* 17(1):25-76.

[5] Liu G. R., Liu M. B. 2003, Smoothed particle hydrodynamics: A meshfree particle method. *World Scientific*, Singapore.

[6] Monaghan J J. 1992, "Smooth particle hydrodynamics," *Annual Review of Astronomy And Astrophysics,* 30:543-574.

[7] Greenhow M, Lin W M. 1983, "Nonlinear free surface effects: Experiments and theory," *Depart Ocean Eng*, Report No.83-19.